

# ANSYS - Wind Turbine Blade


Author: Ben Mullen, Cornell University

## Problem Specification

1. Pre-Analysis & Start-Up
  2. Geometry
  3. Mesh
  4. Physics Setup
  5. Numerical Solution
  6. Numerical Results
  7. Verification & Validation
- Exercises  
Comments

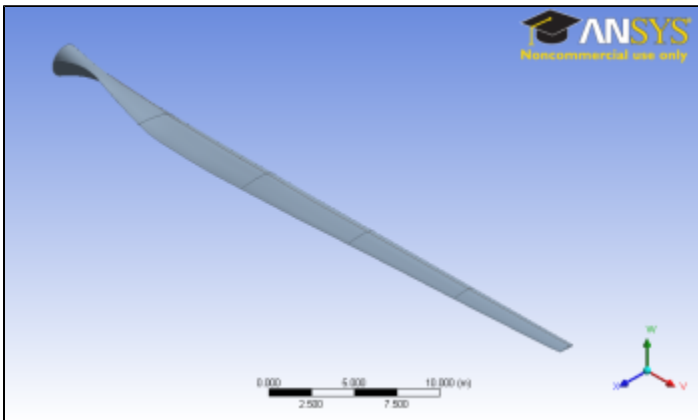
## Wind Turbine Blade

Created using ANSYS 13.0

 This tutorial is not being updated any more. We recommend that you follow [this newer tutorial](#) on fluid-structure analysis of a wind turbine blade. Thank you!

## Problem Specification

This tutorial is based on [this M.Eng project report](#) completed at Cornell University in 2011. In this exercise, we will examine the stresses and deformation of a wind turbine blade under a force load.



[Click here to enlarge image](#) The blade is composed of an outer surface and an inner spar. The spar is 0.02 meters thick and the outer surface is of varying thickness. A table containing the material properties of the two parts is below. Structural steel will be used for both the blade and the spar.

|                      | Spar | Wind Blade          |
|----------------------|------|---------------------|
| Young's Modulus (Pa) | 2e11 | 2e11                |
| Poisson's Ratio      | 0.30 | 0.30                |
| Thickness (m)        | 0.02 | 0.03 - 0.00048485*X |

We are now ready to open ANSYS Workbench

[Go to Step 1: Pre-Analysis & Start-Up](#)

[Go to all ANSYS Learning Modules](#)